

## STEAM CONDENSER'S BODY FLANGE AND COVER FLANGE DESIGN VALIDATION USING FEA

P. M. DESAI<sup>1</sup>, B. C. PATHAK<sup>2</sup>, V. A. PATEL<sup>3</sup> & P. B. RANA<sup>4</sup>

<sup>1</sup>Mechanical Engineering, SVIT, Vasad, Gujarat, India

<sup>2,3,4</sup>Mechanical Engineering, Government Engineering College, Bharuch, Gujarat, India

### ABSTRACT

Weight optimization of body flange and cover flange used in condenser is major area of interest since many years. Most of the researchers have worked on functional ability of bolted joints with gasket under operating conditions. This paper will provide an exposure to design and analysis of body flange & cover flange of steam condenser used in thermal power plant by using FEM approach and validate analytically as per ASME.

**KEYWORDS:** Body Flange, Cover Flange, Finite Element Analysis

### INTRODUCTION

A Flange joint is composed of separate and independent, although inter-related components like flanges, gaskets and bolts, which are assembled by another influence, the assembler. Proper control must be exercised in the selection and application for all these elements to attain a joint that has acceptable leak tightness.

In the case of the ASME, Appendix Y, the bending stress value in three directions longitudinal, radial and tangential can be determined directly using an analytical approach. The equation for the maximum longitudinal hub bending stress given in the code does not indicate the location of this stress but only the magnitude. The radial flange bending stress can either be determined at the bolt circle or at the inside diameter. The tangential Flange bending Stress was determined from the Appendix Y procedure at the inside diameter of the flange and so is compared directly with the FEA predictions. [1]

The analysis of bolted flanged joint with gaskets is due to the nonlinear behavior of the gasket material combined with permanent deformation. The material undergoes permanent deformation under excessive stresses. The degree of elasticity is a function of the compressive stresses which act on the gasket during assembly and after it is put into service. Gasket stiffness has a predominant effect on the behavior of the joint because of its relatively low stiffness. [2]

### DESIGN OF FLAT HEAD

Here the Design of Flange is based on the ASME Codes Section-8, Div-I and the dimensions obtained are

Thickness of shell = 12 mm

Thickness of Body Flange = 102 mm

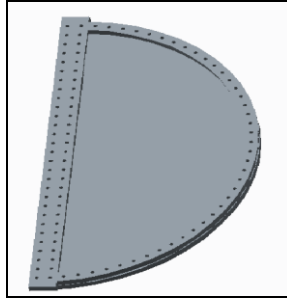
Thickness of Cover Flange = 60 mm

Material for shell, cover flange and body Flange is SA 516 Gr 70.

The dimensions obtained by the calculations are used to model the part by using commercial modeling software.

## FINITE ELEMENT ANALYSIS

For carrying out analysis, 3D model as shown in figure 1 has been prepared in commercial software creo 1.0 using dimension obtained from ASME section-8, Div-I. The model is simplified by removing man holes, nozzle etc. as shown in figure 1.



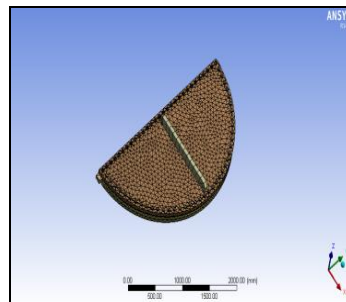
**Figure 1: Symmetry Model of Flange**

Material property for given flange is as shown table 1.

**Table 1: Material Properties**

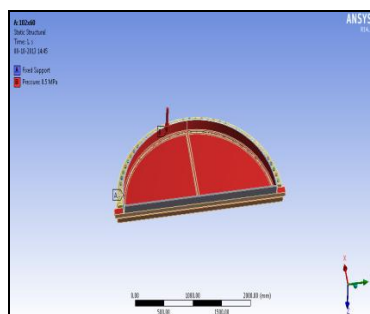
S. No	Mechanical Properties	Value
1	Density	7850 Kg/ m <sup>3</sup>
2	Young's Modulus	195 GPa
3	Poisson's ratio	0.3
4	Yield strength	250 MPa

Model is meshed with tetrahedral elements as shown in figure 2. Total number of nodes and elements are 130847 and 52075 respectively.



**Figure 2: Mesh Model**

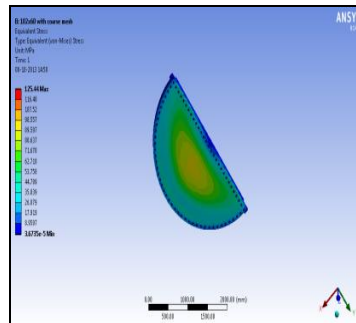
Boundary condition of body flange and cover flange are such that it's periphery is fixed i.e. all degree of freedom is zero. Boundary condition for given analysis is as shown in figure 3.



**Figure 3: Boundary Condition**

## FEM VALIDATION OF FLANGE ASSEMBLY

The initial validation of the simulation results have been carried out by comparing them with results obtained from analytical calculation(ASME Code, Sec-8, DivI). Figure 4 shows the von- Mises stress of cover flange.



**Figure 4: Finite Element Analysis of Cover Flange**

Numerical simulation results are well with in allowable limiting stress 138 MPa. Results reveal that maximum stress value of FE analysis is 125.44 MPa.

## CONCLUSIONS

Finite element analysis is very useful tool for design and analysis. Here finite element analysis is used to validate the design which was carried out from the ASME code. Analytical stress value is 127.9 MPa and simulation stress value is 125.44 MPa for cover flange. The deviation of finite element result and analytical result is approximately 1.9%.

Some design modification can be done as a future work in order to optimize the cover flange for given operating condition.

## REFERENCES

1. M. Abid and D.H.Nash, " A Parametric study of metal to metal contact flanges with optimised geometry for safe stress and no leak condition"-International Journal of Pressure vessels and Piping81, 2004, pp 67-74.
2. M. Murlikrishna, M.S. Shunmugam And N.Siva Prasad, "A study of the sealing performance of bolted flanged joints with gaskets using finite element analysis"- International Journal of Pressure Vessels and Piping 84,2007,pp-349-357
3. ASME Boiler And Pressure Vessel Code, Section-8,Div-1,2010 edition, Rules of construction for Pressure vessel
4. ASME Boiler And Pressure Vessel Code, Section-2, Div-D, 2010 edition, Properties Materials.

